

CAREER EPISODE 3

INTRODUCTION:

Date of Start of Project	10 th February 2017
Date of Completion of Project	1 st April 2017
Location of the Project	Chennai, Tamil Nadu, India
Organization	St. Joseph's Institute of Technology
Project Name	Investigation Analysis of Reducing Drag Force on Passenger Car
Designation & Role	Mechanical Engineer

CE 3.1

This career episode describes my project bearing title **“Investigation Analysis of Reducing Drag Force on Passenger Car”**. This project was carried out to fulfill the requirement of Mechanical Engineering Degree completion.

BACKGROUND:

CE 3.2

Nature of the Project

A few years back when the fuel crisis was not a problem, cars were mainly designed for high-speed maneuverability, comfort, and safety. However, since 2002 with the recent impact of increasing fuel prices the decreasing sales of automobiles have crippled the industry all over the world. This was immediately followed by many questions raised regarding the effect of oil supply shortage on the future of this industry. Numerous resolutions were undoubtedly recommended and countless non-practical resolutions were now assumed thoughtful deliberation. Also, the advancement of electric cars and fuel-cells, other projected methods comprise the incorporation of air-conditioning systems with electronic devices to cut down energy consumption, the redesign of car frames and bodies to reduce the total weight, and the alteration of car physiques to progress the complete aerodynamic physiognomies for better-quality journeying circumstances, dependable triangulation and inferior-energy feasting. These enhancements are also indirectly related to the environment and noise pollution.

In the process of car enterprise, the aerodynamic-smoothness needs to be utterly worked upon. Vehicle design can, usually, only be satisfactory if its drag is abridged to a set mark. Numerous investigators have used CFD methodology to achieve arithmetical replications linked to vehicle-aerodynamics.

This project presents the development process of aerodynamic-holography of the automobile external frame. I performed numerous arithmetical simulations to analyze the pressure, velocity vector field & aerodynamic force forecasting associated with the passenger car. Then, I examined the steadiness of the aerodynamic forces caused by the airflow over the car. Thereafter, I prudently assessed the installation of additional instruments that move way to inferior aerodynamic-drag. Through FLUENT this work implied the $k-\varepsilon$ -turbulence-model in a stable position flow to calculate the flow-field about the car and its additional instruments.

As a matter of fact, it is very uncommon to use $k-\varepsilon$ turbulence model in an iteration-dependent problem. Though, it was the goalmouth of this project to prove the viability of assimilating this rare tactic in this type of computational model. Within a relatively short amount of time, this computational process can estimate the aerodynamics of a car with high accuracy.

CE 3.3

Objective of the Project

The main objective of our project was as follows,

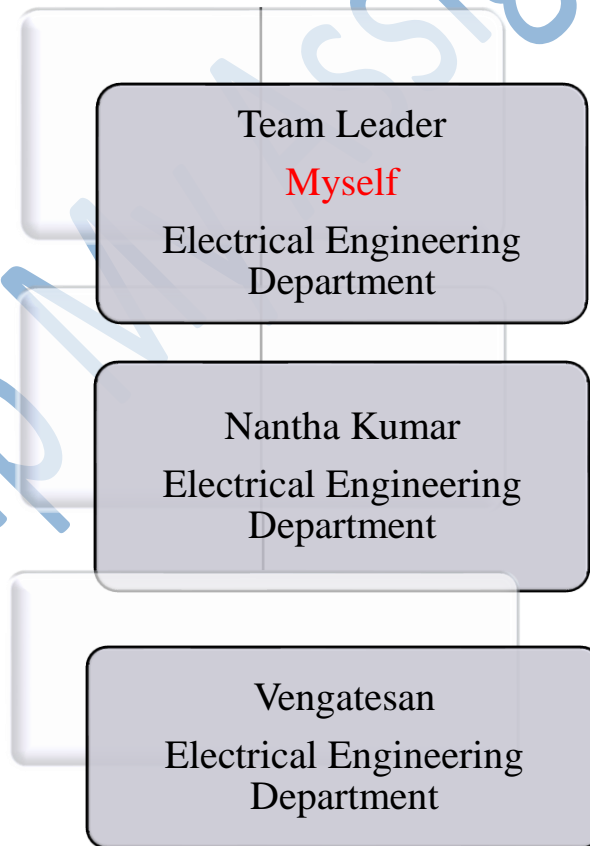
- To improve the fuel efficiency and steadiness of the passenger car
- To reduce drag force by introducing various add-on devices for the passenger car

This project proposed an effective numerical model using the Computational Fluid Dynamics (CFD) to obtain the flow structure around a passenger car with different add-on devices. I built the computational / numerical model of the passenger car and mesh implying A.N.S.Y.S-Fluent which is the C.F.D-solver and was employed in the present work. In this project, I completed the numerical iterations and then aerodynamic data and detailed complicated flow structure were visualized. I developed a model of generic passenger car using solid works, and applied the boundary condition in ANSYS workbench platform, and then testing and simulation were achieved for the appraisal of drag-coefficient for vehicle. In addition to that, the

aerodynamics of the most suitable design of vortex generator (VGs), spoiler, tail plate, and spoiler with VGs were introduced and analyzed the evaluation of drag constant for private car. The accumulation of these additional instruments lessens the drag-coefficient and lift-coefficient in colliding airstream. Circumnavigating the edges somewhat abridged drag in colliding airstream but does not bring about the significant improvements in the aerodynamic efficiency of the passenger car with add-on devices. Hence, the drag force can be reduced by using add-on devices on vehicle, also fuel economy and firmness of a private car can be enhanced meaningfully.

CE 3.4

This profound project was completed along with two other team members. The project Hierarchy is given below.



PERSONAL ENGINEERING ACTIVITY

CE 3.5

I handled all the management related aspects of the project. I managed the reporting of the project progress to be presented to the project supervisor on fortnight basis. I also prepared the project final report and got it passed acceptably from project supervisor.

CE 3.6

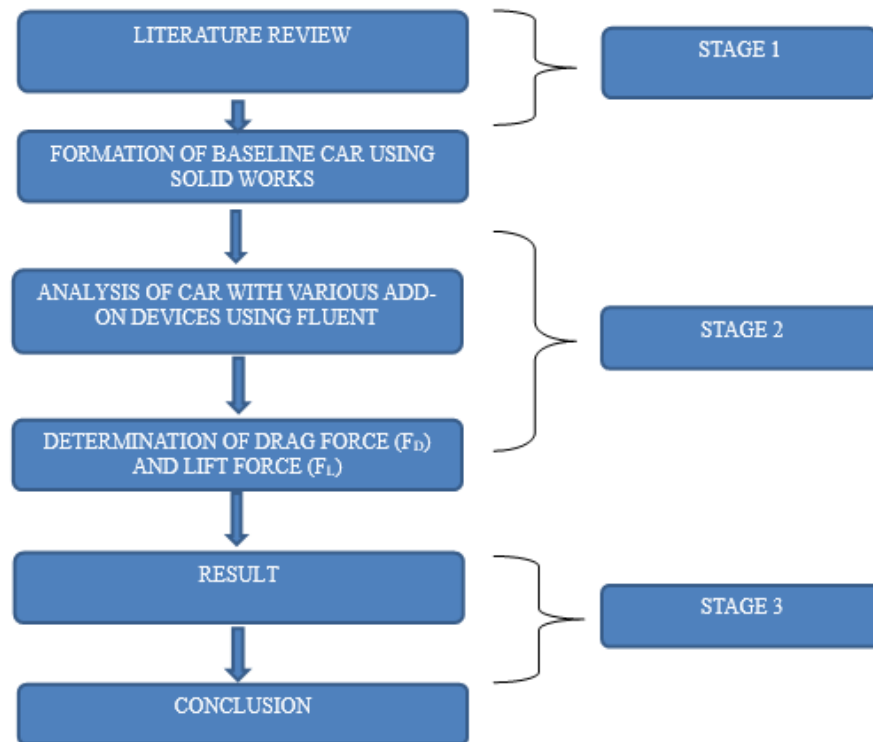
It was evident for me and team to search for the work done in this subject of the project. We did offline search and made use of the university central library. We also searched online and gathered numerous papers and journals to study thoroughly.

CE 3.7

My Roles & Responsibilities

- Researching for the work done in the project scope
- I created and applied the concept
- I parted 3D design model
- I did drag force, lift force, velocity of air when passing a vehicle analysis by using ANSYS
- Engineering design & calculations
- Detailed reporting of the project after completing each milestone
- Drafting final project report

I also presented the project work plan as following.



CE 3.8

Project Methodology

In this project I first prepared a generic model of the passenger car in the Solid works software and this generic model was imported into the A.N.S.Y.S C.F.D to do the simulation of the coefficient of drag and coefficient of lift which is generated in the design module of the ANSYS FLUENT.

After this I generated the meshing on the surface of the passenger car. Aerodynamic assessment of airflow over an article were achieved by implying analytical technique or C.F.D method. Analytical technique of resolving air-flow over an article can be done only for modest flows over modest geometries. If airflow gets complex as inflows over a bluff-body, the flow develops turbulent and it is unbearable to resolve Navier-Stokes and steadiness equations systematically. In order to come up with reasonable solution I used a time-averaged Navier-Stokes equation along with turbulent-models to solve the glitch encompassing Reynolds stress resulting from the time averaging process.

In the present work the I selected $k-\varepsilon$ turbulence-model with non-equilibrium wall function to analyze the flow over the generic passenger car model. This $k-\varepsilon$ turbulence model is very robust, having reasonable computational turn-around time, and widely used by the auto industry.

Steps for Simulation

I as a team leader defined the steps involved in the simulation

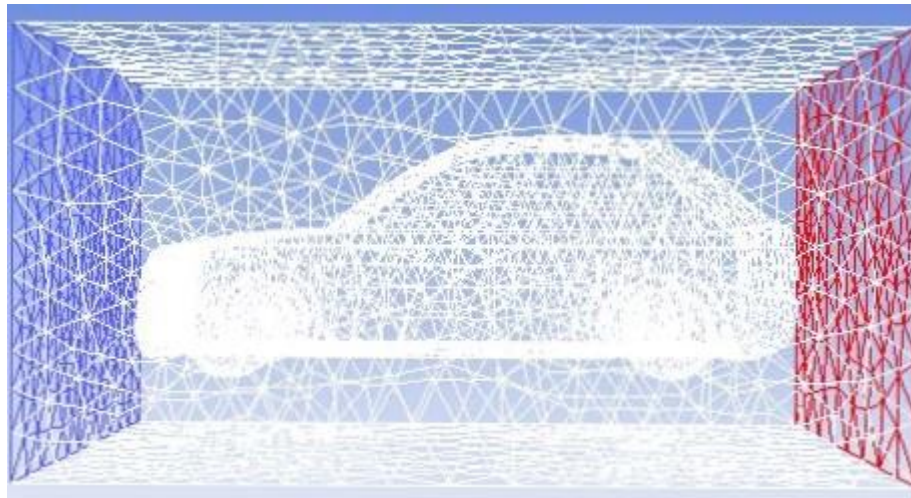
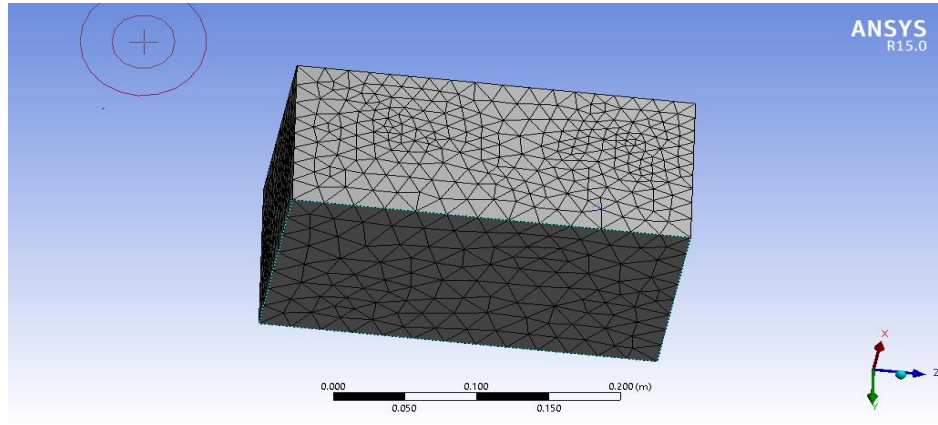
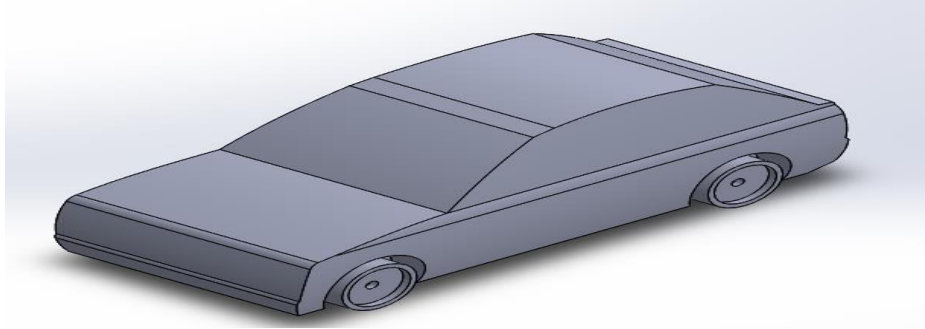
- Select the models of vehicle upon which add-on devices are to be used.
- Formation of passenger car model: design a model in solid works with proper dimensions and parameters.
- Passenger car CFD method and setup: apply the boundary conditions.
- Simulate and test the box type passenger car for drag coefficient and lift coefficient.
- Simulate and test the passenger car without add-on devices for drag coefficient and lift coefficient.
- Simulate and test the passenger car with spoiler for drag coefficient and lift coefficient.
- Simulate and test the passenger car with tail plate & Tail plate for drag coefficient and lift coefficient.
- Impact of add-on devices on fuel economy of passenger car.

CE 3.9

Formation of Box Type Car Model

I designed the model of generic passenger car in solid works. Below figure shows the isometric view of the generic passenger car used in the present CFD simulation. The full size of the generic passenger car was 220mm long, 100mm wide and 90mm high. Thereafter, this model has been analyzed for drag constant and lift coefficient under the ANSYS (FLUENT) module.

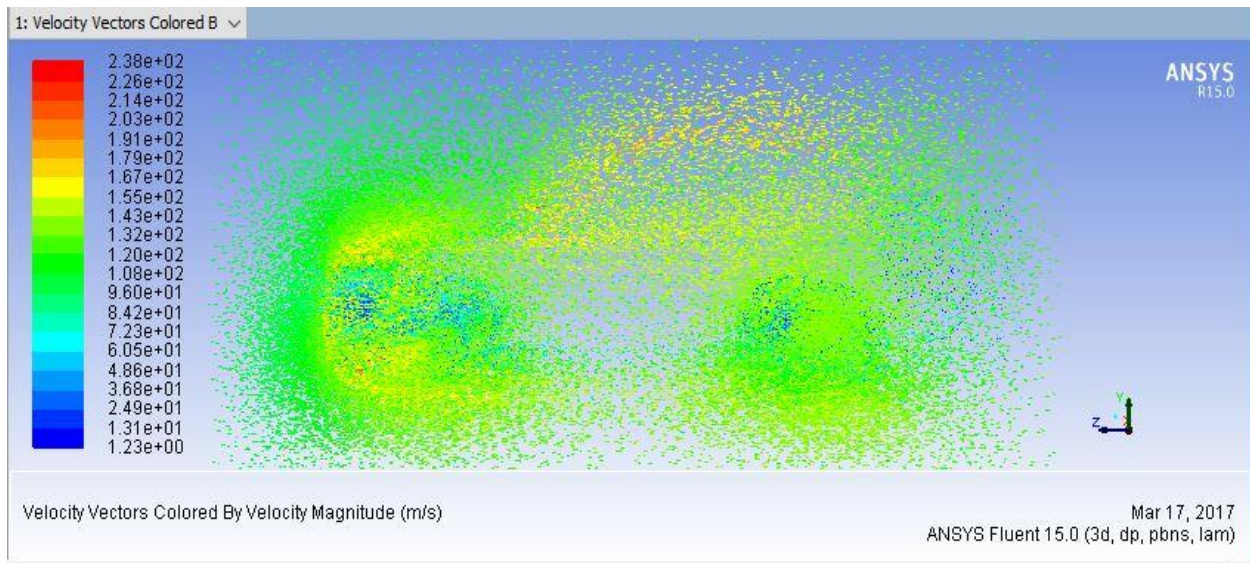
The superficial-mesh of general passenger-car is displayed in below Figure. The tetrahedrons type meshing was produced on its surface. A surface-mesh of active full size is created on the vehicle surface.



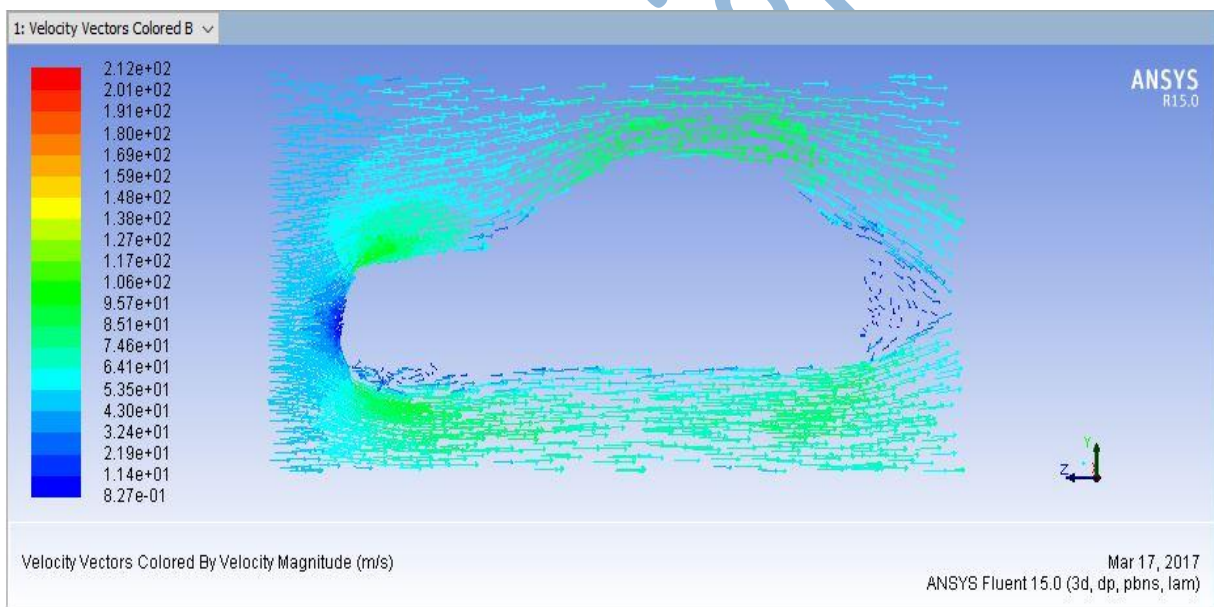
CE 3.10

Simulation & testing of box type car for Drag Coefficient & Lift Coefficient

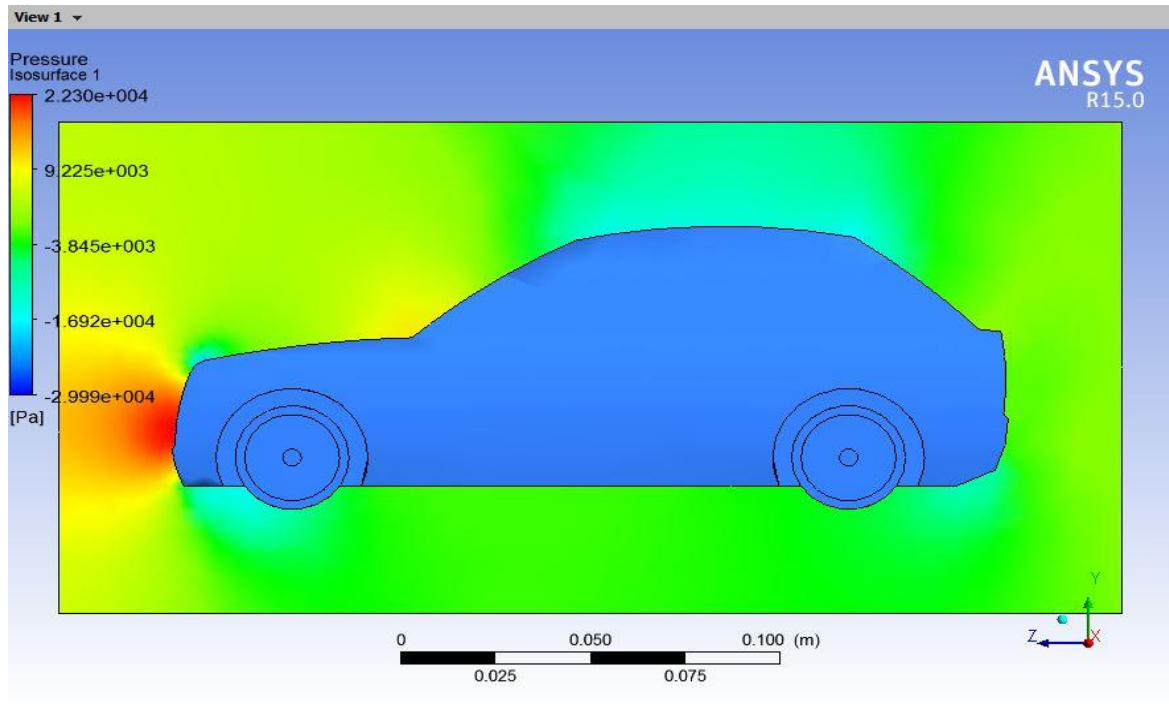
Below pictures shows the velocity contour and number of velocity path lines on surface of baseline model passenger car. It also shows the maximum and minimum velocity occurred on the surface of the baseline model of passenger car.



Velocity Contour



Velocity Vector



Pressure Contour

CE 3.11

Summary

The project played noteworthy role in my academic years.. I gained a lot of practical knowledge about the solid works and 3D simulations.

I presented along with the team the Computational fluid dynamics (CFD) simulations of the stable flow-field round passenger vehicle along with or without additional devices comparing the simulated data to each other. The A.N.S.Y.S Fluent with the $k-\epsilon$ steady model was used by us for the simulations of aero dynamics. In this analysis, the coefficient of drag was reduced up to 84.75% and coefficient of lift was reduced up to 4.43% by the addition of spoiler setup. Hence, the spoiler setup was the effective add-on device to reduce the drag force on vehicle.

The effects of different aerodynamic add-on devices on flow and its structure over a generic passenger car can be analyzed using CFD tactic. The objective was to decrease aerodynamic-drag interim on the car and thus elevate the fuel efficacy of vehicle. Henceforth, the drag-force

can be reduced by using add-on devices on vehicle. So, the fuel economy and steadiness of a passenger car can be enhanced.

Help My Assignment